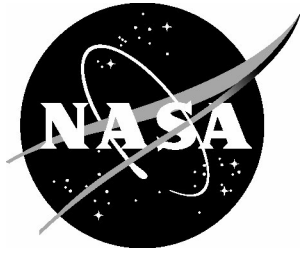


NASA/TM-2003-212651



Piezoelectric Actuator Modeling Using MSC/NASTRAN and MATLAB

Mercedes C. Reaves and Lucas G. Horta
Langley Research Center, Hampton, Virginia

October 2003

The NASA STI Program Office . . . in Profile

Since its founding, NASA has been dedicated to the advancement of aeronautics and space science. The NASA Scientific and Technical Information (STI) Program Office plays a key part in helping NASA maintain this important role.

The NASA STI Program Office is operated by Langley Research Center, the lead center for NASA's scientific and technical information. The NASA STI Program Office provides access to the NASA STI Database, the largest collection of aeronautical and space science STI in the world. The Program Office is also NASA's institutional mechanism for disseminating the results of its research and development activities. These results are published by NASA in the NASA STI Report Series, which includes the following report types:

- **TECHNICAL PUBLICATION.** Reports of completed research or a major significant phase of research that present the results of NASA programs and include extensive data or theoretical analysis. Includes compilations of significant scientific and technical data and information deemed to be of continuing reference value. NASA counterpart of peer-reviewed formal professional papers, but having less stringent limitations on manuscript length and extent of graphic presentations.
- **TECHNICAL MEMORANDUM.** Scientific and technical findings that are preliminary or of specialized interest, e.g., quick release reports, working papers, and bibliographies that contain minimal annotation. Does not contain extensive analysis.
- **CONTRACTOR REPORT.** Scientific and technical findings by NASA-sponsored contractors and grantees.

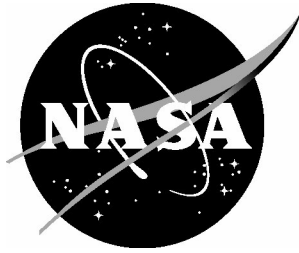
- **CONFERENCE PUBLICATION.** Collected papers from scientific and technical conferences, symposia, seminars, or other meetings sponsored or co-sponsored by NASA.
- **SPECIAL PUBLICATION.** Scientific, technical, or historical information from NASA programs, projects, and missions, often concerned with subjects having substantial public interest.
- **TECHNICAL TRANSLATION.** English-language translations of foreign scientific and technical material pertinent to NASA's mission.

Specialized services that complement the STI Program Office's diverse offerings include creating custom thesauri, building customized databases, organizing and publishing research results ... even providing videos.

For more information about the NASA STI Program Office, see the following:

- Access the NASA STI Program Home Page at [*http://www.sti.nasa.gov*](http://www.sti.nasa.gov)
- E-mail your question via the Internet to [*help@sti.nasa.gov*](mailto:help@sti.nasa.gov)
- Fax your question to the NASA STI Help Desk at (301) 621-0134
- Phone the NASA STI Help Desk at (301) 621-0390
- Write to:
NASA STI Help Desk
NASA Center for AeroSpace Information
7121 Standard Drive
Hanover, MD 21076-1320

NASA/TM-2003-212651



Piezoelectric Actuator Modeling Using MSC/NASTRAN and MATLAB

Mercedes C. Reaves and Lucas G. Horta
Langley Research Center, Hampton, Virginia

National Aeronautics and
Space Administration

Langley Research Center
Hampton, Virginia 23681-2199

October 2003

The use of trademarks or names of manufacturers in the report is for accurate reporting and does not constitute an official endorsement, either expressed or implied, of such products or manufacturers by the National Aeronautics and Space Administration.

Available from:

NASA Center for AeroSpace Information (CASI)
7121 Standard Drive
Hanover, MD 21076-1320
(301) 621-0390

National Technical Information Service (NTIS)
5285 Port Royal Road
Springfield, VA 22161-2171
(703) 605-6000

ABSTRACT

This paper presents a procedure for modeling structures containing piezoelectric actuators using MSC/NASTRAN and MATLAB. The paper describes the utility and functionality of one set of validated modeling tools. The tools described herein use MSC/NASTRAN to model the structure with piezoelectric actuators and a thermally induced strain to model straining of the actuators due to an applied voltage field. MATLAB scripts are used to assemble the dynamic equations and to generate frequency response functions. The application of these tools is discussed using a cantilever aluminum beam with a surface mounted piezoelectric actuator as a sample problem. Software in the form of MSC/NASTRAN DMAP input commands, MATLAB scripts, and a step-by-step procedure to solve the example problem are provided. Analysis results are generated in terms of frequency response functions from deflection and strain data as a function of input voltage to the actuator.

INTRODUCTION

NASA Langley Research Center, Industry, and Academia have been actively studying and developing induced strain actuation devices for aircraft and aerospace applications since the late 1980's¹⁻². Strain actuation produced as a result of a thermal load and piezoelectricity³ involves straining of components due to temperature changes and voltage changes. Piezoelectric materials such as Lead Zirconate Titanate (PZT) ceramics when subjected to an electric field produce mechanical strain or alternately generate an electric charge when subjected to a mechanical strain. This property gives piezoelectric materials the ability to act as actuators or sensors. Using piezoelectric actuators and sensors to form self-controlling and self-monitoring systems to improve performance of aircraft and space structures has attracted interest in the research community.

Numerous studies have been completed on analyses and models for piezo-electrically controlled structures. Some of these studies include: a high-order theory to model composite laminates with surface bonded or embedded piezoelectric sensors by Seely⁴; a three-dimensional finite element code to analyze the mechanical-electrical response of laminated composites containing distributed piezoelectric ceramics developed by Sung Kyu Ha⁵; the use of classical laminate theory to estimate the through-the-thickness strain distribution of composite laminates with embedded actuators by Crawley⁶ and others⁷⁻⁸. Although those analytical techniques showed good correlation with experimental data, few have been incorporated into commercially available codes for general use.

Due to the increased interest in the design of complex structures with piezoelectric actuators and the need for fast and simple implementation of piezoelectric control systems, technology developers are beginning to incorporate or provide the tools to create piezoelectric elements. Freed⁹ developed one and two-dimensional finite elements, which include piezoelectric coupling, for integration into MSC/NASTRAN, however, this option is not currently available. Hauch¹⁰ investigated using ABAQUS electromechanical-coupled finite elements and superelement capabilities for modeling structures with piezoelectric actuators. These advances in the modeling capabilities of

piezoelectric actuators have allowed a number of viable analytical and numerical tools. Reaves¹¹ developed benchmark test article structures to validate techniques for modeling structures containing piezoelectric actuators using commercially available Finite Element Analysis (FEA) packages. This document presents detail information on the implementation of one approach described in reference 11. The use of these tools is presented using a cantilever aluminum beam with surface mounted piezoelectric actuators as a sample case.

The method described herein use MSC/NASTRAN to model the structure with piezoelectric actuators and a thermally induced strain to model expansion/contraction of the actuators due to an applied voltage field. To reduce the number of structural modes needed for an accurate solution, Ritz vectors¹² are appended to the structural modes. MATLAB scripts are used to assemble the dynamic equations and to generate frequency response function at the locations of interest. Software in the form of MSC/NASTRAN DMAP input commands, MATLAB scripts, and a step-by-step procedure to solve the case history example are provided. Frequency response functions from deflection and strain responses to applied input voltage are used for evaluation of the methodology.

PROBLEM FORMULATION

Modeling of a structure with piezoelectric actuators using finite elements can be performed like any conventional finite element model formulation. However, to include piezo effects, elements that expand or contract as voltage is applied, the thermal strain capability within the element formulation needs to be used to evaluate voltage induced internal forces. The governing equation of motion is written as

$$M\ddot{x} + Kx = f_p \quad (1)$$

where f_p represents internal loads due to actuation, M is the mass matrix, K is the stiffness matrix, and x contains node displacements. Induced forces are computed by first defining an artificial coefficient of thermal expansion $\alpha = d_{31} / t$ using the piezoelectric strain to voltage parameter d_{31} and the piezo material thickness t . The second step is to apply a unit change in temperature (or voltage) in NASTRAN to evaluate the resulting deformation T_r . This deformation vector T_r is referred to as a Ritz vector, one Ritz vector is required for each independently actuated piezo. Internal loads are now related to displacements and voltages by $f_p = KT_r \Delta v$. In the reduced order model formulation, mass normalized eigenvectors Φ from an eigenvalue solution of Eq. (1) and a set Ritz vectors T_r can be used as a set of basis vectors for model reduction. With these basis vectors a new coordinate transformation can be defined

$$x = [\Phi \quad T_r] \begin{Bmatrix} z \\ z_r \end{Bmatrix} \quad (2)$$

to transform Eq. (1) to

$$\begin{bmatrix} I & M_r \\ M_r^T & M_{rr} \end{bmatrix} \begin{Bmatrix} \ddot{z} \\ \ddot{z}_r \end{Bmatrix} + \begin{bmatrix} \Lambda & K_r \\ K_r^T & K_{rr} \end{bmatrix} \begin{Bmatrix} z \\ z_r \end{Bmatrix} = \begin{bmatrix} K_r \\ K_{rr} \end{bmatrix} \Delta v \quad (3)$$

where Λ is a diagonal matrix containing the eigenvalues to the homogeneous form of Eq. (1), I is an identity matrix, and all the other matrices are partitioned consistent with the number of basis vectors used. For example, using m eigenvalues and r piezoelectric actuators, the transformation matrix in Eq. (2) has $m+r$ columns and n rows, where n is the number of degrees-of-freedom in the model. Similarly the transformed displacement vector z has m and z_r has r rows. Often, Eq. (3) is sufficient to conduct simulation studies, however, in this form the equation corresponds to a stiff system of differential equations because the transformed mass matrix is ill conditioned. Ill conditioning in this particular case is due to negligible mass values associated with the Ritz basis vectors. To overcome this problem, one can decompose the mass matrix using singular value decomposition such that

$$\begin{bmatrix} I & M_r \\ M_r^T & M_{rr} \end{bmatrix} = U \begin{bmatrix} \sigma & \\ & 0 \end{bmatrix} V^T \quad (4)$$

where U contains the left singular vectors, V contains the right singular vectors, and σ are nonzero singular values. In the numerical implementation of the procedure it is not likely that some of the singular values will be identically equal to zero but values below a certain threshold are considered zero. Furthermore, the number of singular values with numerical values near zero equals the number of piezoelectric actuators placed in the model. Using the right singular vectors, a new transformation matrix can be defined as

$$\begin{Bmatrix} z \\ z_r \end{Bmatrix} = V \begin{Bmatrix} \eta \\ \eta_r \end{Bmatrix} \quad (5)$$

Substituting Eq. (5) into Eq. (3) yields a transformed set of equations

$$\begin{bmatrix} \sigma & \\ & 0 \end{bmatrix} \begin{Bmatrix} \ddot{\eta} \\ \ddot{\eta}_r \end{Bmatrix} + \begin{bmatrix} k & k_r \\ k_r^T & k_{rr} \end{bmatrix} \begin{Bmatrix} \eta \\ \eta_r \end{Bmatrix} = \begin{Bmatrix} b \\ b_r \end{Bmatrix} \Delta v \quad (6)$$

where the bottom set of equations correspond to an algebraic set of equations. Separating the algebraic equation from the differential equations, the resulting properly conditioned set of matrix differential equation is

$$\sigma \ddot{\eta} + (k - k_r k_{rr}^{-1} k_r^T) \eta = (b - k_r k_{rr}^{-1} b_r) \Delta v \quad (7)$$

Time integration of Eq. (7) provides values for η and the solution for η_r is

$$\eta_r = -k_{rr}^{-1} k_r^T \eta + k_{rr}^{-1} b_r \Delta v \quad (8)$$

To recover the response in physical coordinates, results from Eqs. (7) and (8) are substituted into Eqs. (5) and (2). For MATLAB implementation all the equations are transformed into state space form and then integrated using the MATLAB built-in integration software.

EXAMPLE PROBLEM

A cantilevered aluminum beam 2.75" x 16" x .04", Figure 1, with one piezoelectric actuator bonded near the root is used in this study. A Flex-Patch piezoelectric actuator, developed and fabricated at NASA LaRC, is selected for the application. The Flex-Patch consists of a 3"x1.75"x.008" Morgan Matrox PZT-5A¹³ piezoceramic encapsulated using a polymer film. The PZT-5A piezoceramic mechanical and electro-mechanical properties are listed in Table 1. MSC/NASTRAN is used to model the structure; Figure 2 shows a finite element representation of the cantilevered beam indicating the location of the piezoelectric actuator and strain sensor output used in the present study. The model contains 396 quadrilateral elements and 450 nodes. PCOMP cards in MSC/NASTRAN are used to specify the properties of the composite lay-up for the actuator and aluminum substrate.

ANALYTICAL TOOLS IMPLEMENTATION

The procedure to generate the analytical model of the cantilever beam involves the following steps: 1) construction of a FEM based on the test article and Ritz vectors generation, 2) eigenvectors generation and mass and stiffness matrices calculation to assemble the reduced order model, 3) execution of MATLAB scripts to assemble the dynamic equation and to generate frequency response function at the locations of interest. These steps are described in more detail in the following sections.

Structural FEM and static Ritz vector generation - MSC/NASTRAN offers no piezoelectric coupled-field elements capability from which to model smart structures directly. Rather, the analogy between piezoelectric strain and thermally induced strain, is used to allow temperature changes to model piezoelectric voltage actuation. Piezoelectric coefficients characterizing the actuator are input as thermal expansion coefficients (CTE's) associated with standard elements. In this study the model treats both the actuator and the structures substrates as plies of an integrated laminated plate. PCOMP cards in MSC/NASTRAN are used to specify the properties of the composite lay-up and the applied voltage is modeled as a thermal load. TEMP cards identify the locations at which thermal loads (voltage) is applied. MAT1 cards representing the actuators include non-zero thermal expansion coefficients. The thermal expansion coefficients in the remaining structure must be zero to ensure thermal expansion loads are generated only at the piezoelectric actuator locations. Appendix 1 shows a listing of the MSC/NASTRAN input data deck '**almq2.dat**', for the cantilevered beam. For brevity, MSC/NASTRAN grid points and element information are not included in the listing. The execution of this file will generate the Ritz vector (static deflection and/or strain data for the beam) using a thermal load equivalent to 1 volt. The DMAP sequence '**static.dmap**'

incorporated in the MSC/NASTRAN input data deck stores the static solution vector in a binary formatted output file '**S101SV1.OUT4**'. In applications with multiple actuators each actuator load and output request are to be included in individual SUBCASES.

Eigenvectors and reduced model generation - After computing Ritz vectors, a general eigenvalue/eigenvector MSC/NASTRAN solution containing special structural matrix transformation routines (DMAP) combine the Ritz vectors with eigenvectors and calculates the mass and stiffness matrices in Eq. (3) needed to assemble a reduced order model. NASTRAN commands for the cantilever beam example are listed in the file named '**almq2e.dat**' and the DMAP sequence is listed in '**genkm_v2.dmap**' (Appendix 2). The '**almq2e.dat**' file uses the file '**S101SV1.OUT4**' as input and generates two ASCII formatted output files: '**almq2e.asc**' and '**almq2e.pch**' containing the reduced mass/stiffness matrix and the displacements and/or strains for the structural modes including the Ritz vectors, respectively. Output observation points are for displacements at nodes 348 and 349 and strain for element 116. To facilitate importing the matrices to a MATLAB environment the file '**almq2e.asc**' needs to be edited to remove lines starting with the one containing "eigensum" up to but not including the line containing "genk". The first line in the file should contain 'genk' and must not have blank lines in the beginning of the file.

Dynamic equation assembly and results generation - MATLAB scripts are required to assemble the dynamic equations and generate frequency response functions at the observation points. The MATLAB script '**makemodel.m**' used in the cantilever beam case study is listed in Appendix 3. The script creates space state model matrices based on the number of inputs (number of actuators) and MSC/NASTRAN output request (element strain and/or nodal displacement). The script uses as input the files generated and edited in the previous step: '**almq2e.asc**' and '**almq2e.pch**'. The MATLAB scripts are problem specific depending on the type of output requests, but are easy to modify.

A flow chart summarizing the complete procedure to solve the cantilevered aluminum beam problem is shown in Figure 3.

EXAMPLE PROBLEM RESULTS

The frequency response functions for input voltage, tip displacement, and strain gage data are generated from analysis. NASTRAN analysis results using the modeling technique described herein are shown in Figure 4 for the first four beam/actuator mode shape deformations with corresponding frequencies of 5.68, 33.59, 60.24 and 91.21 Hz. To examine the input/output relationship of the system with the actuator, Figure 5 shows a comparison of the frequency response function magnitude (top) and phase (bottom) from the beam tip displacement to the piezo-actuator input voltage. Furthermore, Figure 6 shows results from the nearly collocated strain gage.

SUMMARY

A procedure for modeling structures containing piezoelectric actuators using MSC/NASTRAN and MATLAB is presented. The procedure describing how to use these tools is presented using a cantilever aluminum beam with surface mounted piezoelectric actuators as example problem. Software in the form of MSC/NASTRAN DMAP input commands, MATLAB scripts, and a step-by-step procedure to solve the example problem are provided. Analysis results are generated in terms of frequency response functions from deflection and strain data to input voltage to the actuator.

REFERENCES

1. Crawley E.F., "Intelligent Structures for Aerospace: A Technology Overview Assessment". AIAA Journal, Vol. 32, No. 8, August 1994.
2. Bailey, T. L., and Hubbard, J.E., "Distributed Piezoelectric-Polymer Active Vibration Control of a Cantilever Beam". Journal of Guidance, Control, and Dynamics, Vol.8, Sept.-Oct. 1985, pp. 605-611.
3. Crawley, E. F. and de Luis J., "Use of Piezoelectric actuators as Elements of Intelligent Structures". AIAA Journal, Vol. 25, No. 10, 1987, pp. 1373-1385.
4. Seeley, C. E. and Chattopadhyay, A., "Analysis of Smart Composite Structures Including Debonding". Ph. D. Dissertation, Arizona State University, May 1997.
5. Ha, S. K., Keilers, C., Chang, Fu-Kuo, "Finite Element Analysis of Composite Structures Containing Distributed Piezoceramic Sensors and Actuators". AIAA Journal, Vol. 30, No. 3, March 1992.
6. Crawley, E.F., and Anderson, E. H., "Detailed Models of Piezoelectric Actuation of Beams". Journal of Intelligent Material Systems and Structures, Vol. 1, January 1990, pp. 4-25.
7. Lam, K.Y., Peng, X. Q., Liu G.R. and Reddy, J.N., "A Finite-element Model for Piezoelectric Composite Laminates". Smart Materials and Structures 6 (1997) 583-591.
8. Won, C.C., Sulla, J.L., Sparks, D.W., Belvin, W.K., "Application of Piezoelectric Devices to Vibration Suppression". Journal of Guidance, Control, and Dynamics, Volume 17, Number 6, Pages 1333-1338.
9. Freed, B.D, Babuska, V., "Finite Element Modeling of Composite Piezoelectric Structures with MSC/NASTRAN". SPIE 4th Annual Symposium on Smart Structures and Materials, San Diego, CA. March 1997. Paper 3041-60.

10. Hauch, R. M., “ABAQUS Smart Structure Modeling Methods”. AMS-TN-95-02, February 21, 1995.
11. Reaves, Mercedes C. and Horta, Lucas, G., “ Test Cases for Modeling and Validation of Structures with Piezoelectric Actuators”. 42nd AIAA/ASME/ASCE/AHS/ASC Structures Dynamics and Materials Conference, April 16-19,2001. AIAA-2001-1466.
12. Sandridge, C. A. and Haftka, R. T., “ Accuracy of Eigenvalue Derivatives from Reduced-Order Structural Models”. Journal of Guidance, Control and Dynamics, Vol. 12, No. 6, Nov.-Dec. 1989.
13. Berlincourt, D. and Krueger, H.A., “Properties of Morgan Electro Ceramic Ceramics”. Technical Publication TP-226. Morgan Electro Ceramics, 2000.
www.morgan_electroceramics.com/techpubl.html

Table 1. Material properties of PZT-5A piezoceramic and T300/976 graphite/epoxy composite

	PZT-5A	T300/976
Modulus of elasticity (lb _f /in ²)		
E ₁	1.0E+7	2.17E+7
E ₂		1.305E+6
Poisson's ratio		
v	0.3	0.3
Shear Modulus (lb _f /in ²)		
G ₁₂	3.82E+6	1.03E+6
G _{1Z}		1.03E+6
G _{2Z}		363600
Density, (lb _f - sec ² /in ⁴)		
ρ	7.16E-4	1.49E-4
Piezoelectric constant, (in/Volt)		
d ₃₁	6.73E-9	
Electrical permittivity, (farads/in)		
ξ ^δ	4.2E-10	

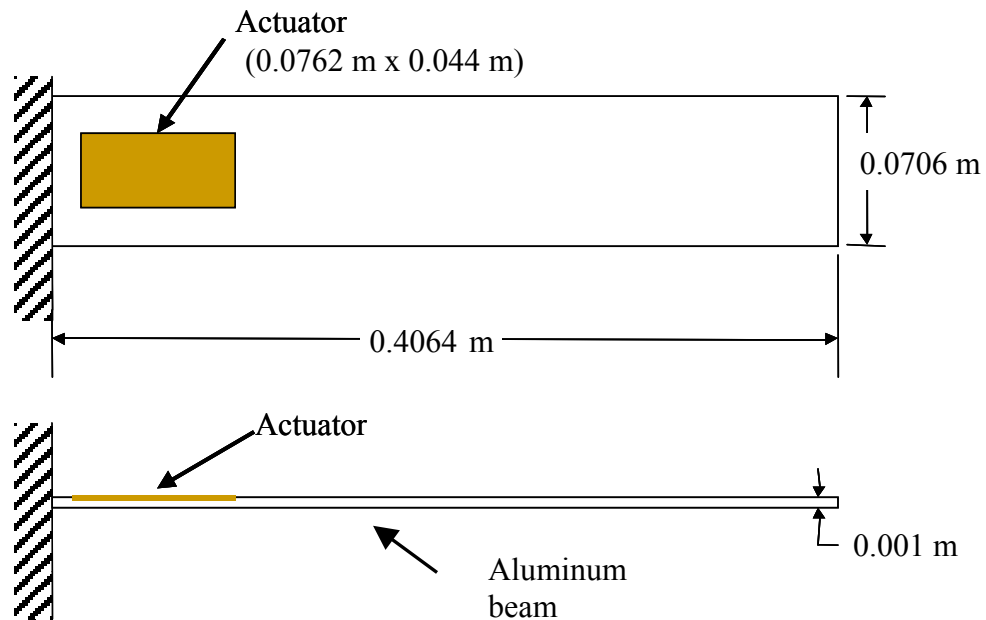
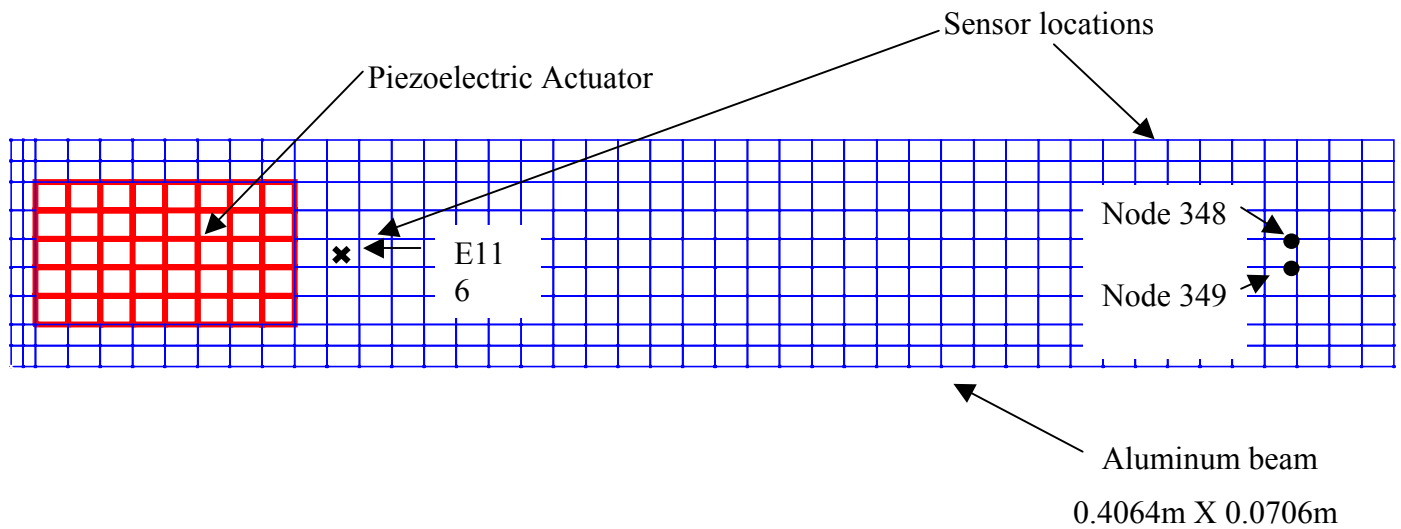


Figure 1. Cantilevered aluminum beam with piezoelectric actuator.



E116 – strain sensor output location

Nodes 348, 349 – displacement output location

Figure 2. Aluminum beam with piezoelectric actuator finite element mesh.

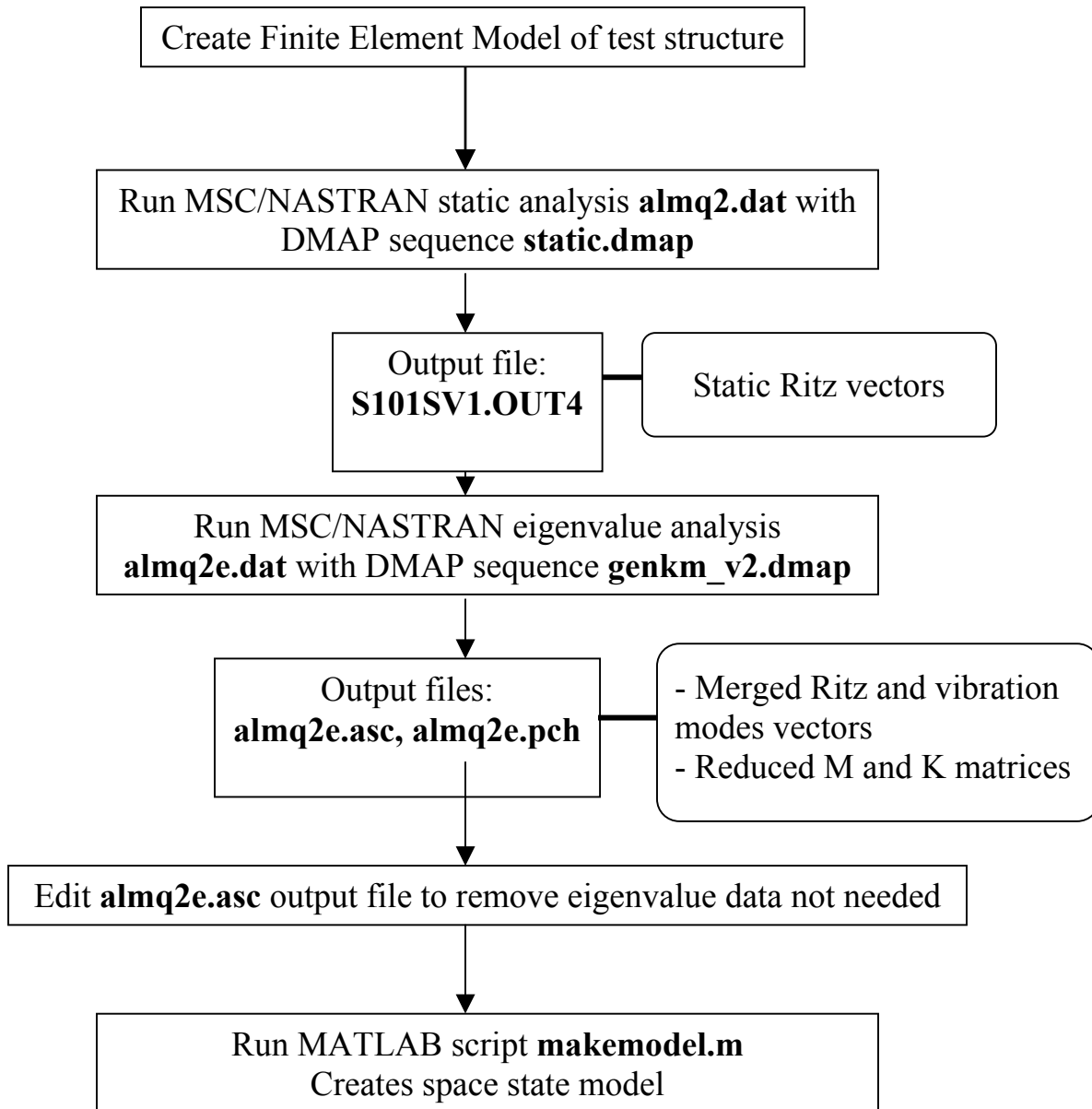


Figure 3. Steps to generate analytical model of aluminum beam with piezoelectric actuator.

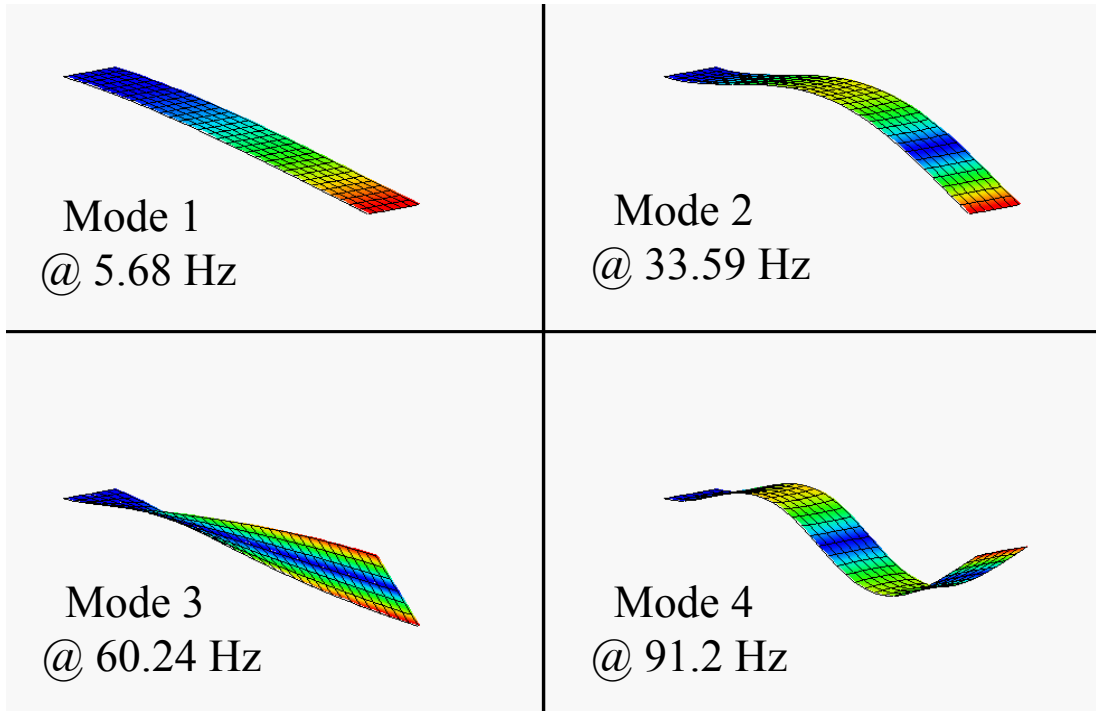


Figure 4. Mode Shapes of aluminum beam with piezoelectric actuator.

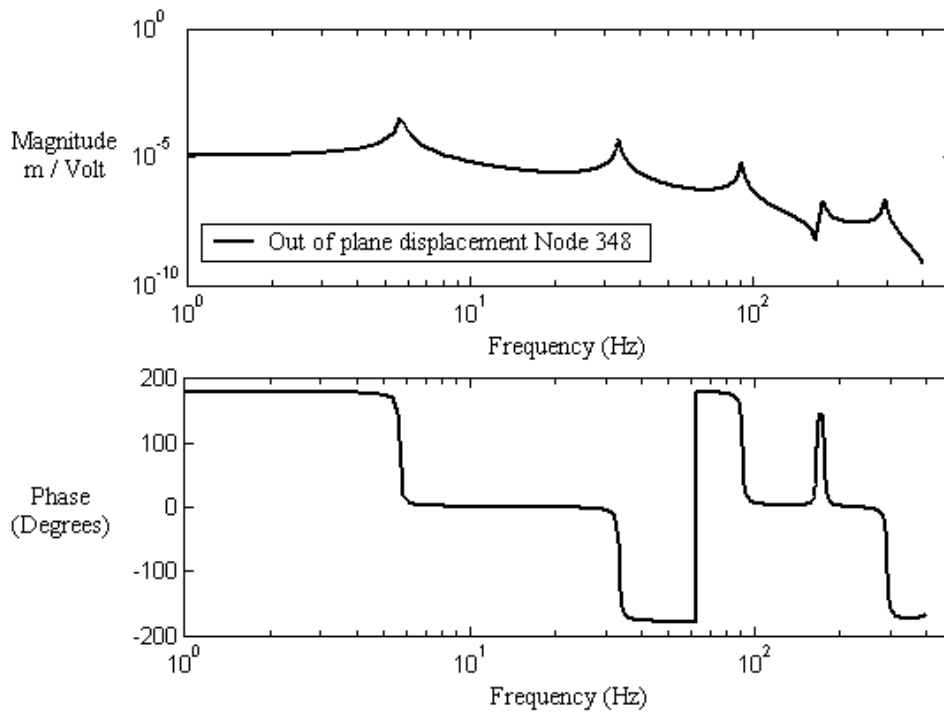


Figure 5. FRF of aluminum beam tip displacement as a function of piezoelectric actuator input voltage.

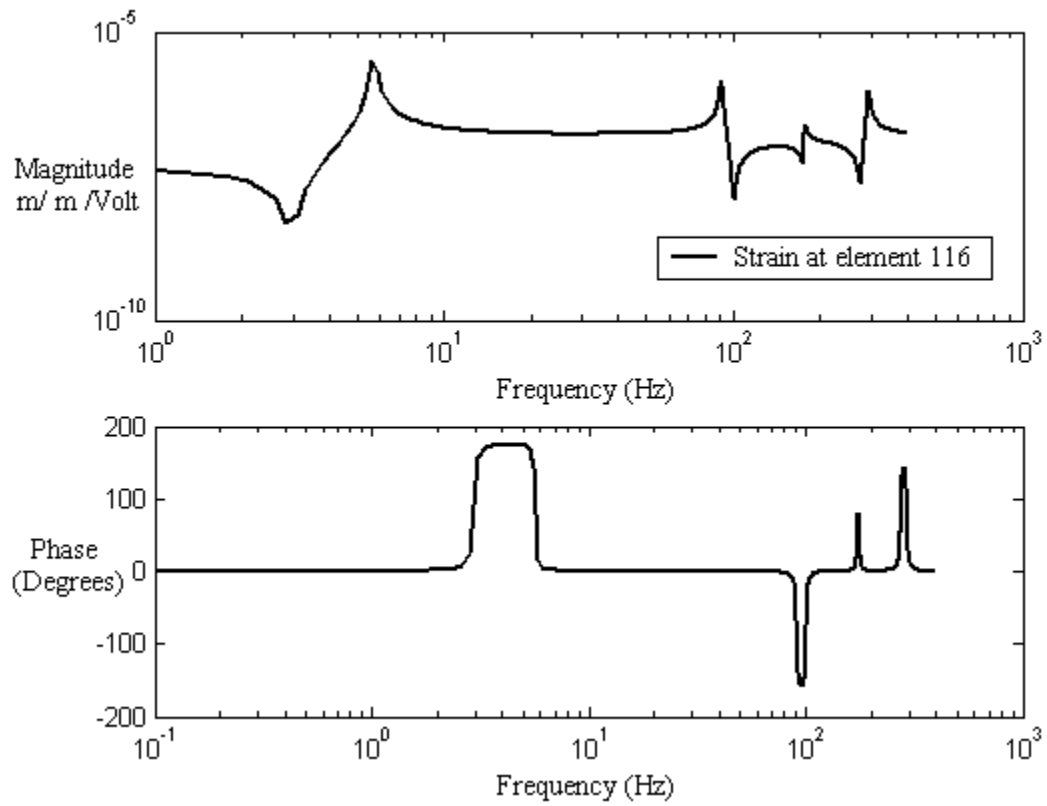


Figure 6. FRF of aluminum beam strain as a function of piezoelectric actuator input voltage.

Appendix 1

The following NASTRAN input data deck listing is contained in the **almq2.dat** file. The deck is set up to generate the Ritz vector corresponding to a single piezoactuator mounted on the structure. TEMP cards identify the locations at which thermal loads (voltage) is applied. The DMAP **static.dmap** incorporated in the data deck by the ‘**include**’ command stores static solution vectors in the output file '**S101SV1.OUT4**' specified in the ASSIGN statement.

almq2.dat data deck:

```
ASSIGN OUTPUT4='S101SV1.OUT4' UNIT=15 DELETE
INIT MASTER(S)
ID PIEZOELECTRIC QUAD ELEMENTS
SOL 101
$
compile sestatic,souin=mscsou,nolist,noref$
include 'static.dmap'
$
  TIME = 60
CEND
TITLE = NASTRAN FILE TRANSLATOR -- UNITS = SI
$ GLOBAL CASE
  SPC = 1
  TEMP = 1
$ Define set of elements for strain output
SET 4 =116
$ Define set of nodes for displacement output
SET 5 =348,349
DISPLACEMENT(PRINT)=5
STRAIN(PRINT,FIBER)=4
$
BEGIN BULK
$
PARAM AUTOSPC YES
PARAM POST -2
PARAM,K6ROT,100.
GRID 1 0 7.14E-3 5.75E-2 0.00000 0
GRID 2 0 7.14E-3 4.87E-2 0.00000 0
GRID 3 0 7.14E-3 3.98E-2 0.00000 0
.
.
.
CQUAD4 396 2 552 513 519 555-90.0000
.
.
```


\$ Property card definition for aluminum beam substrate

PCOMP 2-5.08E-4 0.00000 0.00000 0.00000 +
 + 2 1.27E-5 0.00000YES 2 9.91E-4 0.00000YES +
 + 2 1.27E-5 0.00000YES

\$ Property card definition for aluminum beam substrate and piezoactuator lay-up

PCOMP 5-5.08E-4 0.00000 0.00000 0.00000 +
 + 2 1.27E-5 0.00000YES 2 9.91E-4 0.00000YES +
 + 2 1.27E-5 0.00000YES 3 1.27E-5 0.00000YES +
 + 3 2.54E-5 0.00000YES 4 2.03E-4 0.00000YES +
 + 3 2.54E-5 0.00000YES 3 1.27E-5 0.00000YES

\$ Aluminum material properties

MAT1 2 7.2E+10 0.33333 2758. 0.0

\$Kapton film material properties

MAT1 3 2.76E+9 0.45000 1358. 0.0

\$Piezo ceramic material properties, non-zero CTE

MAT1 4 6.9E+10 0.31000 7700. **8.42E-7**

\$

SPC1 1 123456 504THRU 506

SPC1 1 123456 519THRU 523

SPC1 1 123456 554THRU 555

\$ Unit thermal load (voltage) applied to actuator

TEMP 1 1 1.000 2 1.000 3 1.000
 TEMP 1 4 1.000 5 1.000 6 1.000
 TEMP 1 7 1.000 8 1.000 9 1.000
 TEMP 1 10 1.000 11 1.000 12 1.000
 TEMP 1 13 1.000 14 1.000 15 1.000
 TEMP 1 16 1.000 17 1.000 18 1.000
 TEMP 1 19 1.000 20 1.000 21 1.000
 TEMP 1 22 1.000 23 1.000 24 1.000
 TEMP 1 25 1.000 26 1.000 27 1.000
 TEMP 1 28 1.000 29 1.000 30 1.000
 TEMP 1 31 1.000 32 1.000 33 1.000
 TEMP 1 34 1.000 35 1.000 36 1.000
 TEMP 1 37 1.000 38 1.000 39 1.000
 TEMP 1 40 1.000 41 1.000 42 1.000
 TEMP 1 43 1.000 44 1.000 45 1.000
 TEMP 1 46 1.000 47 1.000 48 1.000
 TEMP 1 49 1.000 50 1.000 51 1.000
 TEMP 1 52 1.000 53 1.000 54 1.000
 TEMPD 1 0.0

ENDDATA

Below is a listing of the 'static.dmap' DMAP:

```
$compile sestatic, souin=mscsou, nolist, noref $
$
$ Following DMAP was coded for MSC/NASTRAN and verified to
$ work in version 2001.0.1 for the current application
$ MSC says this will work for versions 68.2,69, 69.1 or 70
$ This DMAP stores static solution vectors in an output file
$ 'S101SV1.OUT4' specified in the ASSIGN statement.
$ If the FEM does not contain rigid body modes (or mechanisms),
$ then the L-set d.o.f. are written to the output file.
$ If there are rigid body modes (a SUPORT card should be included
$ in the Bulk Data deck), then the T-set d.o.f. are written
$ to the output file.
$
alter 'CALL SUPER3' (1,0)
$alter 685 $ after static solution SSG3
$ If there is no R-set, write the L-set vectors to file
if (norset < 0) then $
output4 ul,,,-1/15/1 $
$ Print L-set solution vector
$ matprn ul// $
endif $
$
$ If the R-set exists, write the T-set vectors to file
$alter 822 $ After SDR1 module
if (norset <> -1) then $
$ Partition the G-set solution, UGVS, into the T-set vectors
if (nogset = notset) then $ no partitions if g-set=t-set
output4 ug,,,-1/15/1 $
$ matprn ug// $
else $
upartn uset,ug/tugvs,,,'g/'t/'o'/1 $
$ matprn tugvs// $ T-set solution displacement vector
output4 tugvs,,,-1/15/1 $
endif $ end of "if (nogset=notset)"
endif $ end of "if (norset <> -1 )"
$
endalter
```

Appendix 2

The following NASTRAN input data deck listing is contained in the **almq2e.dat** file. The executive deck is set up to generate vibration modes. The DMAP sequence **genkm_v2.dmap** incorporated in the data deck by the 'include' command combines the Ritz vectors and eigenvectors and calculates the mass and stiffness matrices needed to assemble a reduced order model. The **almq2e.dat** file uses the file **S101SV1.OUT4** generated by executing **almq2.dat** as input and generates two ASCII format output files: **almq2e.asc** and **almq2e.pch** containing the reduced mass and stiffness matrix and the displacements and/or strains for the structural modes including the Ritz vectors respectively.

```
ASSIGN INPUTT4='S101SV1.OUT4' UNIT=15
ASSIGN OUTPUT4='almq2e.asc' UNIT=16 FORM=FORMATTED DELETE
ID ALBEAM WITH ACTUATOR,PROB
SOL 103
include 'genkm_v2.dmap'
  TIME = 60
CEND
TITLE = NASTRAN FILE TRANSLATOR -- UNITS = SI
$ GLOBAL CASE
  SPC = 1
METHOD=25
$ Output requests are for displacements at nodes 348 and 349
$ and strain for element 116.
$
SET 4 =116
SET 5 =348,349
DISPLACEMENT(PUNCH)=5
STRAIN(PUNCH,FIBER)=4
$
PARTN=5
$
BEGIN BULK
EIGRL,25,,,10
$
PARAM,K6ROT,100.
PARAM AUTOSPC YES
PARAM POST -2
GRID 1 0 7.14E-3 5.75E-2 0.00000 0
```

The rest of the data deck is identical to the listing included in 'almq2.dat' in Appendix 1. TEMP cards are ignored in an eigenvalue analysis.

The following is a listing of the 'genkm_v2.dmap' DMAP:

```
$MSC/NASTRAN DMAP
$ this dmap converted to V69 rjb 1/14/98
$This dmap was verified to work for version 2001.0.1
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$ This DMAP accepts user selected static solution vectors from an
$ input file. The projection of these vectors onto a subspace
$ perpendicular to the subspace of mass weighted normal modes
$ is computed. These new static vectors are appended to the
$ calculated normal modes, and generalized stiffness and mass
$ matrices are formed. The static partition of these generalized
$ matrices are placed on an ASCII output file. Also written to
$ the ASCII output file are: a normal modes summary matrix,
$ and user selected static vectors in external sort order.
$
malter 'CALL.*IFPL.*DMINDX' $ after dmindx is formed
call dbstore dmi,dmindx,,,/99/99/' /0 $
$
compile moders souin=mscsou, nolist, noref $
alter 'LAMA,OEIGS'(1,1) $ after REIGL and OFP modules
$ NEIGV = number of eigenvalues computed
$ NSV = number of columns in static vector matrix STATV
$ STATV is either an L-set or T-set dimensioned static vector matrix
inputt4 /statv,,,/1/15/-1/1 $ read matrix of static vectors
$
$ Matrix SSV is an optional input. Matrix SSV is used to select
$ which static vectors will be used in the following analysis.
$ If matrix SSV is not input, then all the static vectors in the
$ inputt4 file will be used in the analysis.
$
$ DMIIN module is required to read the DMI bulk data entries.
$ The name of the input DMI data block is SSV.
$
type parm,,i,n,indxok
call dbfetch /dmi,dmindx,,,/99/99/0/0/s,indxok
message //'indxok = 'indxok $
dmiin dmi,dmindx/ssv,,,,,,,,/s,n,yesv $
if (yesv) then $ If matrix SSV is input perform following operations
$ Postmultiply STATV by SSV to select the desired static vectors.
$ The resulting vectors are stored in DSV.
mpyad statv,ssv,/dsv $
$ matprn dsv// $
$ Let DSV overwrite STATV
equivx dsv/statv/-1 $ DSV is renamed to STATV
$ matprn statv// $
endif $ Above "IF (yesv)" group only used if SSV exists in bulk data
```


Appendix 3

The following MATLAB script is used to create space state model matrices based on the number of inputs (number of actuators) and MSC/NASTRAN output request (element strain and/or nodal displacement). **Caution:** The MATLAB scripts are problem specific depending on the type of output requests, but easy to modify.

```
makemodel.mat
%*****
% Script file MAKEMODEL reads NASTRAN Files      *
% with the reduced mass, stiffness, mode shapes, and Ritz vectors      *
%                                           *
% A state space model of the system is constructed with input in terms *
% of voltages to the piezoactuators and output as desired      *
%                                           *
% Created by L.G. Horta and M.C. Reaves      *
% NASA Langley 8-01-01      *
%                                           *
%*****
%
Fname2=input(' Enter NASTRAN ascii FileName for reduced mass and stiffness (CR=default):= ','s');
if isempty(Fname2); Fname2='almq2e.asc';disp(Fname2);end

[FID,MESSAGE] = fopen(Fname2,'rt');

[Stiff,Mass]=readMK(FID);    % Reading reduced mass and stiffness matrices

Fname2=input(' Enter NASTRAN Punch FileName (CR=default):= ','s');
if isempty(Fname2); Fname2='almq2e.pch';disp(Fname2);end
[FID,MESSAGE]=fopen(Fname2,'rt');

nq=input(' Number of Output nodes (CR=default):= ');
if isempty(nq); nq=2;disp([' nq = ' num2str(nq)]);end
nm=input(' Number of Modes in File (CR=default):= ');
if isempty(nm); nm=11;disp([' nm = ' num2str(nm)]);end
ne=input(' Number of Strain ELEMENTS (CR=default):= ');
if isempty(ne); ne=1;disp([' ne = ' num2str(ne)]);end
Nplies=input(' Number of Plies per element (CR=default):= ');
if isempty(Nplies); Nplies=3;disp([' Nplies = ' num2str(Nplies)]);end
PlyNum=input(' Ply number where sensor data is measured (CR=default):= ');
if isempty(PlyNum); PlyNum=3;disp([' PlyNum = ' num2str(PlyNum)]);end
NR=input(' Enter Number of Ritz-Vectors (CR=default):= ');
if isempty(NR); NR=1;disp([' NR = ' num2str(NR)]);end

[eigen,Shape]=readShape(FID,nm,nq);    % read node information
[Strain]=readply(FID,Nplies,ne,PlyNum,nm);    % read element information
fclose(FID);

%*****
%                               End of Read Punch File
%*****
%
%   construct State Space Model
%
```

```

ShapeStrain=[Shape; Strain];           % Stack output matrices
[A,B,C,D,ModelDesc]=ConstructABCD70301(NR,Mass,Stiff,ShapeStrain);
%
% Output order is defined by NASTRAN File node order followed by element info
%
% For Example: Node =[ 1x 1y 1z 2x 2y 2z ...nqx nqy nqz Element 1 ....]

SYS=ss(A,B,C,D);
[Nout,junk]=size(C);
Nfpts=1600;
freq=linspace(.1,400,Nfpts)';
System=ss(A,B,C,D);
[mag,phase]=bode(System,freq*2*pi);
Mag=reshape(mag,[Nout*NR Nfpts]);      % Stored Columnwise
Phase=reshape(phase,[Nout*NR Nfpts]);
for j=1:NR
    for l=1:Nout
        subplot(2,1,1),semilogy(freq,Mag((j-1)*Nout+1,:));
        xlabel(' Freq (Hz)');ylabel(' Mag');
        title([' Input No. := ' num2str(j) ' Output No. := ' num2str(l)])
        subplot(2,1,2),plot(freq,Phase((j-1)*Nout+1,:));
        xlabel(' Freq. (Hz)'); ylabel(' Phase');
        disp(' Hit Carriage return to see next plot')
        pause
    end
end
end

```

readMK.mat

```

function [Stiff,Mass]=readMK(FID)
%
% Function to read ASCII File from NASTRAN
% with reduced MASS and Stiffness Matrices
%
% The File must be 81 columns wide
%
% [Stiff,MASS]=readMK(FID)
%
% Input Information
% FID = NASTRAN ASCII file name pointer (81 col. wide)
%
% Output Information
% Mass = Reduced Mass Matrix
% Stiff = Reduced Stiffness Matrix
%
% Written by Lucas G. Horta 01252001

if FID >= 0;
    LINE=fgets(FID);
    NN=str2num(LINE(1:20));      % Two Elements NCOL, NROW
    Mass=zeros(NN(2),NN(1));
    Stiff=zeros(NN(2),NN(1));
    for j=1:NN(1);
        LINE2=fgets(FID);

```



```

        NN2=str2num(LINE2(1:20));    % Two elements COL, RowPos
        Mat=[];
        TT=fscanf(FID,'%g',NN(1));
        TSpace1=fgets(FID);
        Stiff(:,j)=TT;
    end
    LINE2=fgets(FID);
    LINE3=fgets(FID);
    LINE4=fgets(FID);
    for j=1:NN(1);
        LINE2=fgets(FID);
        NN2=str2num(LINE2(1:20));    % Two elements COL, RowPos
        Mat=[];
        TT=fscanf(FID,'%g',NN(1));
        TSpace=fgets(FID);
        Mass(:,j)=TT;
    end
    fclose(FID);
else
    disp(' File not found')
end

```

readShape.mat

```

function [eigen,Shape]=readShape(FID,nmodes,nq)
%
%   Function to read PUNCHFILE from NASTRAN
%   with shape information
%
%   The File must be 81 columns wide
%
%   [eigen,Shape]=readShape(FID,nmodes,nq)
%
% Input Information
% Filename      = NASTRAN Punch file name (ASCII) (81 col. wide)
% nmodes        = Number of modes
% nq            = Number of output nodes
%
% Output Information
% eigen         = Eigenvalue squared (Rad/sec)
% Shape         = Shape Vector size (ngpt*6 x nmodes)
%
% Written by Horta-Reaves 7-03-01
if FID >= 0;
for ijk=1:nmodes
    for jj=1:6
        LINE=fgets(FID);        %Header
    end
    LINE7=fgets(FID);           % Eigenvalue ID
    eigen(ijk)=str2num(LINE7(14:30));
    tem=[];
    for l=1:nq
        LINE1=fgets(FID);
        [nn,mm]=size(LINE1);
        pxyz=str2num(LINE1(20:mm));
    end
end

```

```

        pxyz=pxyz(1:3);
        LINE3=fgets(FID);
    %    Rotation DOF removed
    %    [nn,mm]=size(LINE3);
    %    txyz=str2num(LINE3(20:mm));
    %    txyz=txyz(1:3);
    %    tem=[tem pxyz txyz];
            tem=[tem pxyz];
        end
        Shape(:,ijk)=tem';
    end
else
    disp(' File not found')
end

```

readply.mat

```

function [Strain]=readply(FID,Nplies,ne,PlyNum,nm)
%
%    Supporting file to read NASTRAN punch file info from composite elements
%
%    [Strain]=readply(FID,Nplies,ne,PLyNum,nm)
%
% Input
%    FID   = File handle
%    Nplies = Number of plies
%    PlyNum = Ply number where output is recovered
%    ne    = Number of strain elements
%    nm    = Number of modes
%
% Output
%    Strain= [ne x nm] with strain influence in the Normal-1 Direction for all elements
%
%
% Read strain on PLyNum NORMAL-1 Direction
% Created 8-01-01 by L. Horta
Strain=zeros(ne,nm);
Numberoflines=(PlyNum-1)*4;
for ijk=1:nm
    for jj=1:8
        LINE=fgets(FID);          %Header
    end
    tems=[];
    for l=1:ne
        % Read strain on PlyNum, NORMAL-1 Direction
        if Numberoflines > 0;
            for z1=1:Numberoflines
                LINE=fgets(FID);
            end
        end
        LINE1=fgets(FID);
        [nn,mm]=size(LINE1);
        exy=str2num(LINE1(20:mm));
    % Select NORMAL-1 Direction

```

```

% To get normal-2 change the number 2 to 3 (See NASTRAN Output convention)
    exy=exy(2);
    tems=[tems exy];
    for z2=1:(Nplies-PlyNum)*4+3
        LINE=fgets(FID);      % Remaining plies info not needed
    end
end
Strain(:,ijk)=tems';
end

```

ConstructABCD70301.mat

```

function [A,B,C,D,ModelDesc]=ConstructABCD70301(NR,Mass,Stiff,Shape)
%
%   Function to construct A,B,C,D, matrices for a piezoelectric actuated
%   system
%
%   [A,B,C,D]=ConstructABCD70301(NR,Mass,Stiff,Shape)
%
% Input Information
% NR      = Number of Ritz Vectors
% Mass     = Mass Matrix
% Stiff    = Stiffness Matrix
%
% Output Information
% eigen    = Eigenvalue squared (Rad/sec)
% Shape    = Shape Vector size (Number of Outputs x nmodes)
% ModelDesc = Text Describing output definition
%
%   Written by Horta-Reaves 7-03-01 (Mod 8-01-01)
%
%   construct State Space Model with Ritz vector and mode shape info
%
[NM,Njunk]=size(Mass);
%
%   Reduction of Algebraic equation for Ritz Vectors
%
I1=[1:NM-NR]';
I2=[NM-NR+1:NM]';
[u,s,v]=svd(Mass);
Index=abs(diag(s))>1e-10;
NMnew=sum(Index);
Knew=u'*Stiff*v;
Bnew=u'*Stiff(:,I2);
K11=Knew(I1,I1);
K12=Knew(I1,I2);
K22=Knew(I2,I2);
b1=Bnew(I1,:);
b2=Bnew(I2,:);
K22in=inv(K22);
CP=Shape*v;
C1_P=CP(:,I1);

```

```

C2_P=CP(:,I2);
% Mod 3-28-01
[Ns1,junk]=size(Shape);
D_P=zeros(Ns1,NR);

Mred=s(Index,Index);

Kred=inv(Mred)*(K11-K12*K22in*K12');
Bred=inv(Mred)*(b1-K12*K22in*b2);
Cred_P=C1_P-C2_P*K22in*K12';
Dred_P=D_P+C2_P*K22in*b2;

[Sae,Ar]=eig(Kred);
Br=Sae'*Bred;
Cr_P=Cred_P*Sae;
Dr_P=Dred_P;
disp(' Default damping value is 1 %');
SSS=input(' Do you want to change the damping value for each mode (1=yes,0=no) ');
Damp=2*sqrt(Ar)*eye(NMnew,NMnew)*.01;          % Percent Damping
%
%      Modifications to add damping to individual modes
%      L.G. Horta 6-21-00
%
if SSS == 1;
    DampValue=zeros(NMnew,1);
    for III=1:NMnew;
        disp([' Frequency =' num2str(sqrt(diag(Ar(III,III)))/(2*pi)) ' (Hz)'])
        DampValue(III)=input('Enter damping value in % = ');
    end
    Damp=2*sqrt(Ar)*diag(DampValue)/100;
end

A=[zeros(NMnew,NMnew) eye(NMnew,NMnew); -Ar -Damp];
B=[zeros(NMnew,NR); Br];

C_1=[Cr_P zeros(Ns1,NMnew)];
C_2=[-Cr_P*Ar -Cr_P*Damp];
D_1=Dred_P;
D_2=Cr_P*Br+Dr_P;

C=[C_1;C_2];
D=[D_1;D_2];
%
%  Output Definition
%
ModelDesc=strvcat([' Number of output states is equal to ' num2str(Ns1*2)],...
    [' The first ' num2str(Ns1) ' are translations'],...
    [' From ' num2str(Ns1+1) ' to 'num2str(Ns1*2) ' are
    accel. ']);
disp(ModelDesc)

```

REPORT DOCUMENTATION PAGE				Form Approved OMB No. 0704-0188	
<p>The public reporting burden for this collection of information is estimated to average 1 hour per response, including the time for reviewing instructions, searching existing data sources, gathering and maintaining the data needed, and completing and reviewing the collection of information. Send comments regarding this burden estimate or any other aspect of this collection of information, including suggestions for reducing this burden, to Department of Defense, Washington Headquarters Services, Directorate for Information Operations and Reports (0704-0188), 1215 Jefferson Davis Highway, Suite 1204, Arlington, VA 22202-4302. Respondents should be aware that notwithstanding any other provision of law, no person shall be subject to any penalty for failing to comply with a collection of information if it does not display a currently valid OMB control number.</p> <p>PLEASE DO NOT RETURN YOUR FORM TO THE ABOVE ADDRESS.</p>					
1. REPORT DATE (DD-MM-YYYY) 01- 10 - 2003		2. REPORT TYPE Technical Memorandum		3. DATES COVERED (From - To)	
4. TITLE AND SUBTITLE Piezoelectric Actuator Modeling Using MSC/NASTRAN and MATLAB			5a. CONTRACT NUMBER		
			5b. GRANT NUMBER		
			5c. PROGRAM ELEMENT NUMBER		
6. AUTHOR(S) Reaves, Mercedes C.; Horta, Lucas G.			5d. PROJECT NUMBER		
			5e. TASK NUMBER		
			5f. WORK UNIT NUMBER 23-762-40-31		
7. PERFORMING ORGANIZATION NAME(S) AND ADDRESS(ES) NASA Langley Research Center Hampton, VA 23681-2199			8. PERFORMING ORGANIZATION REPORT NUMBER L-19005		
9. SPONSORING/MONITORING AGENCY NAME(S) AND ADDRESS(ES) National Aeronautics and Space Administration Washington, DC 20546-0001			10. SPONSOR/MONITOR'S ACRONYM(S) NASA		
			11. SPONSOR/MONITOR'S REPORT NUMBER(S) NASA/TM-2003-212651		
12. DISTRIBUTION/AVAILABILITY STATEMENT Unclassified - Unlimited Subject Category 39 Availability: NASA CASI (301) 621-0390 Distribution: Standard					
13. SUPPLEMENTARY NOTES An electronic version can be found at http://techreports.larc.nasa.gov/ltrs/ or http://ntrs.nasa.gov					
14. ABSTRACT This paper presents a procedure for modeling structures containing piezoelectric actuators using MSC/NASTRAN and MATLAB. The paper describes the utility and functionality of one set of validated modeling tools. The tools described herein use MSC/NASTRAN to model the structure with piezoelectric actuators and a thermally induced strain to model straining of the actuators due to an applied voltage field. MATLAB scripts are used to assemble the dynamic equations and to generate frequency response functions. The application of these tools is discussed using a cantilever aluminum beam with a surface mounted piezoelectric actuator as a sample problem. Software in the form of MSC/NASTRAN DMAP input commands, MATLAB scripts, and a step-by-step procedure to solve the example problem are provided. Analysis results are generated in terms of frequency response functions from deflection and strain data as a function of input voltage to the actuator.					
15. SUBJECT TERMS MATLAB; MSC/NASTRAN; Finite element; Modeling; Piezoelectric actuators; Procedure; Structures					
16. SECURITY CLASSIFICATION OF:			17. LIMITATION OF ABSTRACT	18. NUMBER OF PAGES	19a. NAME OF RESPONSIBLE PERSON
a. REPORT	b. ABSTRACT	c. THIS PAGE			STI Help Desk (email: help@sti.nasa.gov)
U	U	U	UU	29	19b. TELEPHONE NUMBER (Include area code) (301) 621-0390